



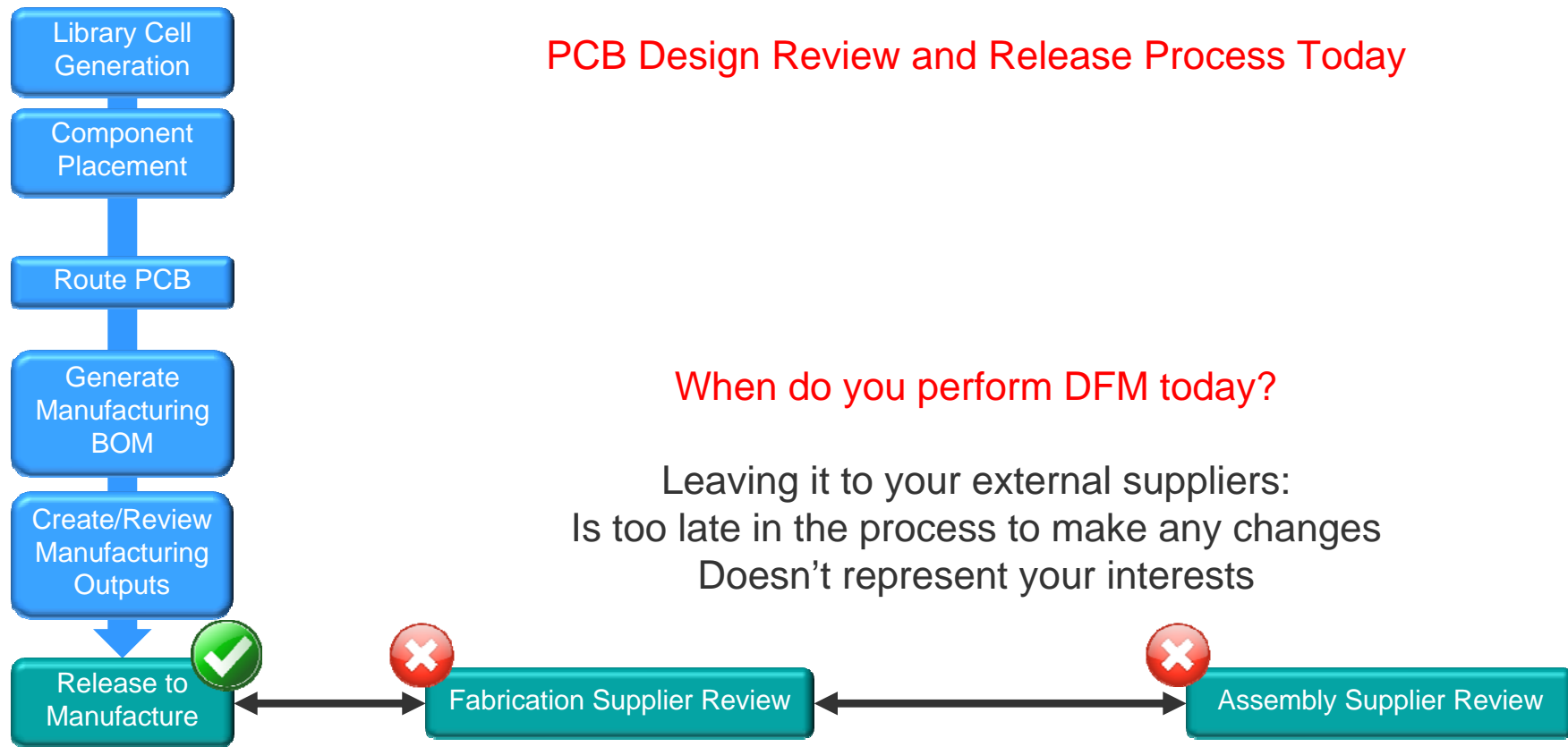
# How Your Suppliers Look at Your Design

Ammar Abusham  
Sr. Applications Engineer  
April, 2017

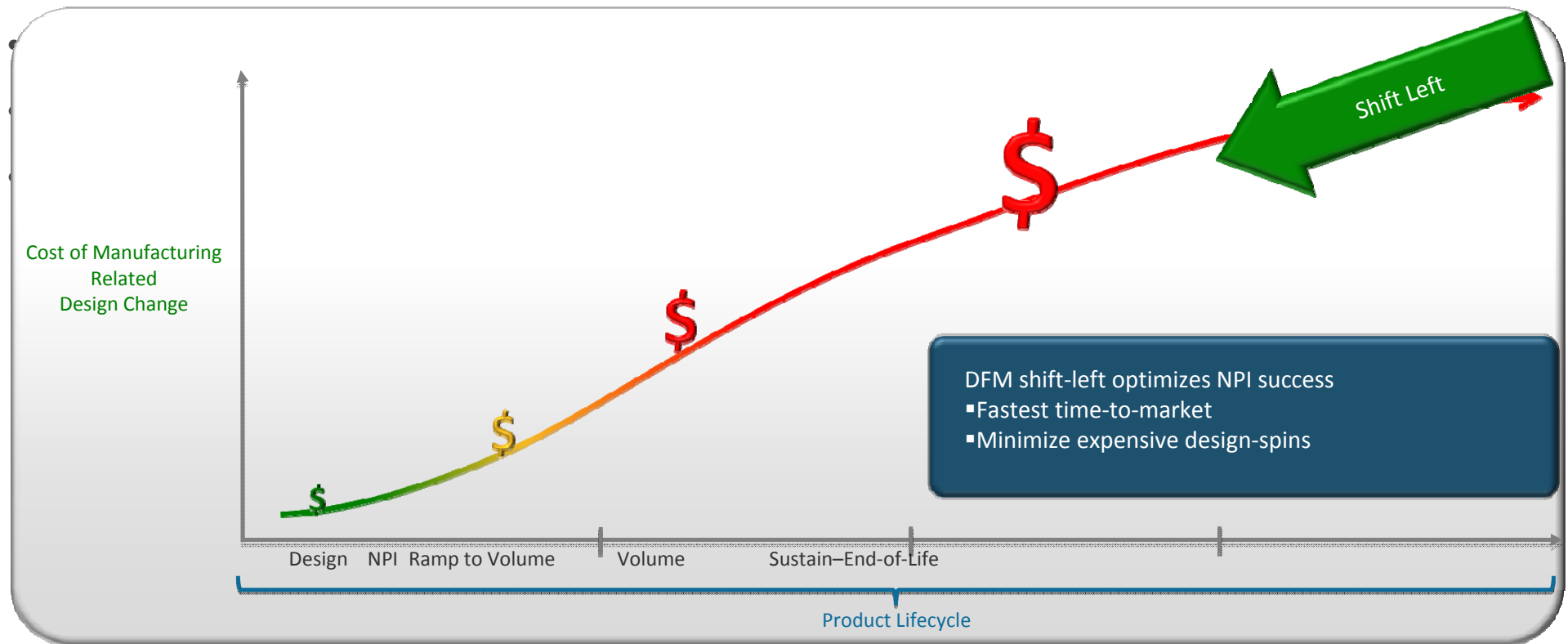
# How Your Suppliers Look at Your Design

- We See Things Differently
- A Fabricator's View
- A Contract Manufacturer's View
- DFM Complements DRC
- Summary

# We See Things Differently



# We See Things Differently



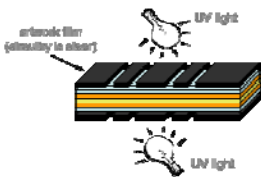
## We See Things Differently

### DFM Interests

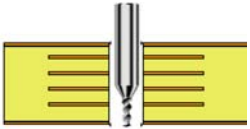
Subject	Design Organization	External Manufacturer
Product costs	Minimize costs	Maximize profit
DFM trade-offs	Wants control of decisions	Wants control of decisions
Form factor	Miniaturization	Maximum manufacturing yield
Design revision spins	Wants to minimize/eliminate	Makes money off of each spin
PCB design integrity	Wants PCB built as designed	Wants/needs to edit PCB design for manufacturing processes
Communication	Expects full disclosure of all edits	Will communicate as required
Component sourcing	Wants control for quality reasons	Wants control for costs/profit reasons
Product reliability	Of utmost concern	Of little concern

# A Fabricator's View

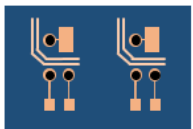
## ■ Inner Layer Processing



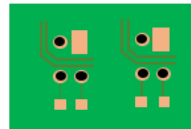
## ■ Drill



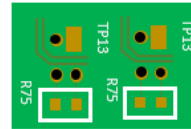
## ■ Outer Layer Processing



## ■ Soldermask



## ■ Silkscreen



## ■ Electrical Test

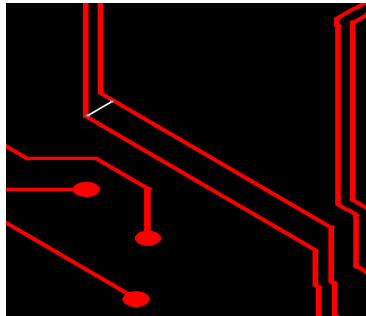


# A Fabricator's View

## Inner Layer Processing



Slivers can cause repeat defects due to photo-resist flaking



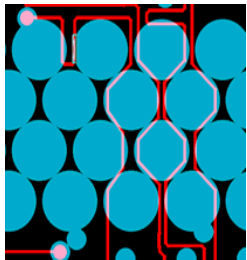
Differential Pairs should maintain the same gap distance and not diverge



Manufacturing process tolerances can cause a same net short, potentially affecting circuit behavior

# A Fabricator's View

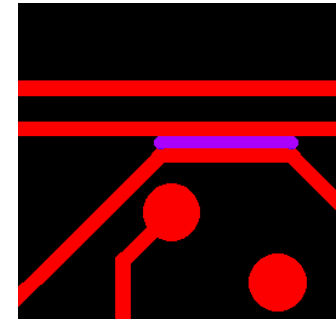
## Outer Layer Processing



Interruptions in the ground plane that prevent electrical continuity disrupt current flow effectively lengthening the return path



Necking down can create voltage bottleneck, and signal discontinuity



Close spacing over an extended length affects yield and therefore costs

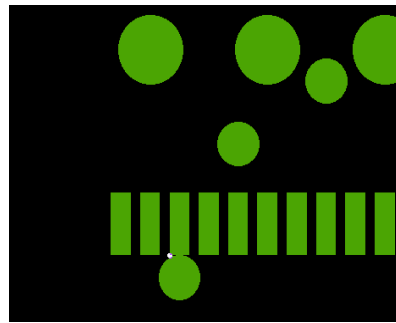


# A Fabricator's View

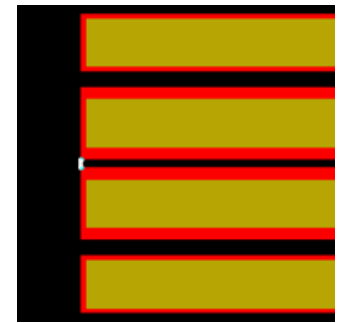
## Soldermask



Circuits close to a pad must be fully covered by mask to avoid solder bridging during the assembly process



Very thin Soldermask will fail to adhere to the PCB



Gangmask is required for fine-pitch devices

# A Fabricator's View

## Silkscreen



Silkscreen too close to pads could jeopardize solder adhesion



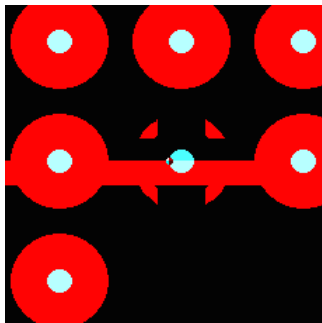
Silkscreen on SMD pads can interfere with soldering



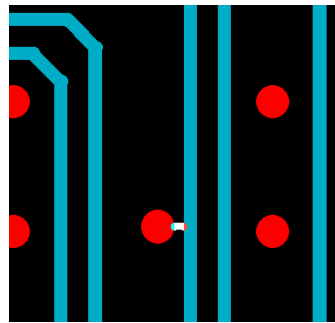
Non-legible silkscreen can be problematic during rework

# A Fabricator's View

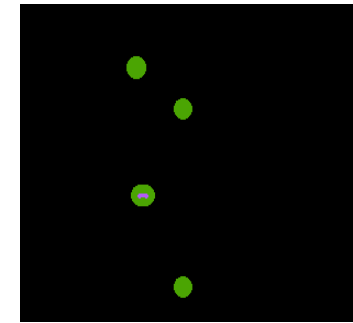
## Drill



Starved thermals prevent proper heat containment, affecting quality of via solder connection



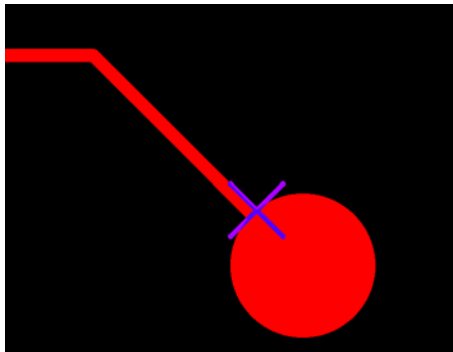
Backdrill diameter should be distant from traces



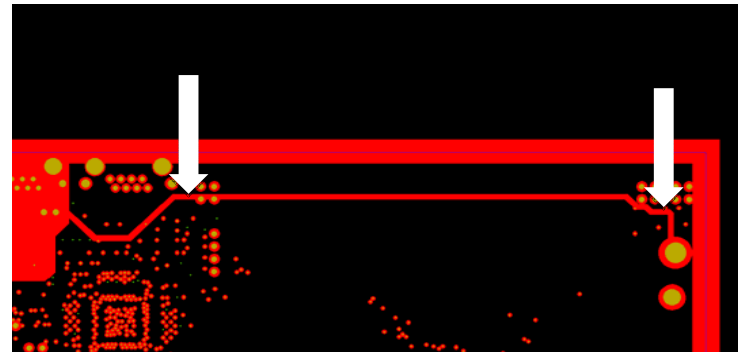
Duplicate drill hit may not drill or plate properly

# A Fabricator's View

## Electrical Test



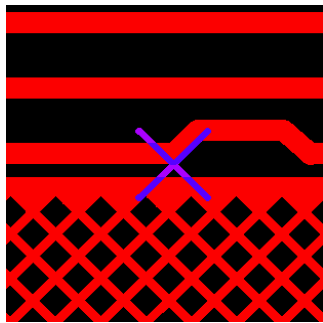
Non-Intentional shorts can cause scrap



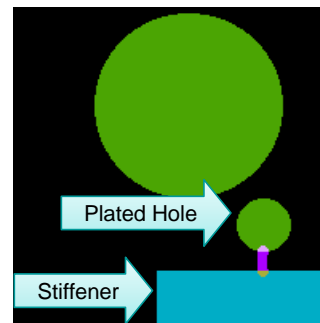
Broken GRND as result of anti-etch on negative plane

# A Fabricator's View

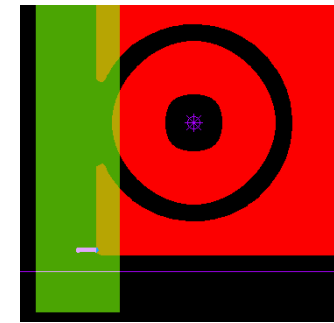
## Flex and rigid-flex



Conductors should be perpendicular across the entire bend area



Plated holes too close to a stiffener can lead to cracks in the barrel



Rigid area copper close to interface area can cause copper to crack

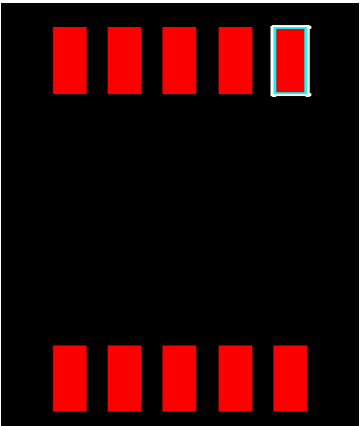
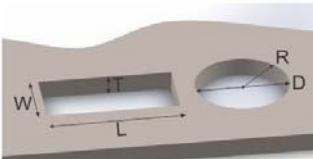
## A Contract Manufacturer's View

- Stencil
- SMT placement
- Manual placement
- AOI/AXI
- Reflow, wave & selective soldering
- Test & repair

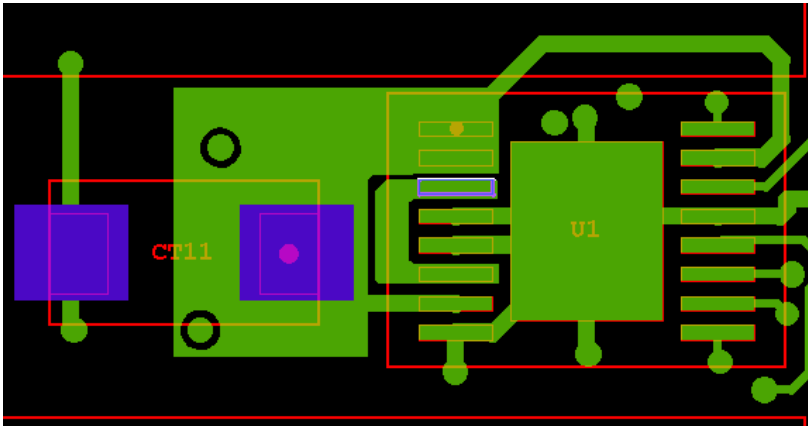
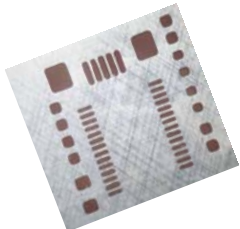


# A Contract Manufacturer's View

## Stencil



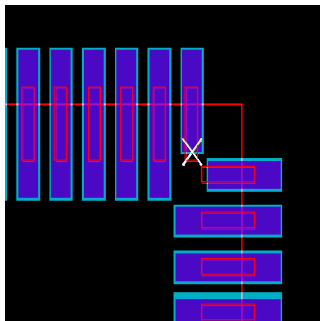
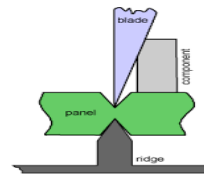
Aspect Ratio of solder paste can greatly affect the deposit of solder paste



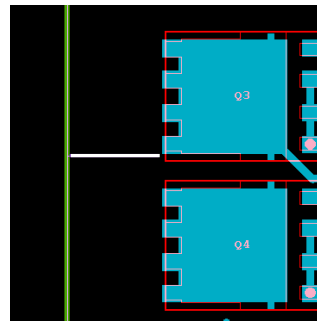
Unexposed component footprint on solder paste layer can result in missing openings in the physical stencil

# A Contract Manufacturer's View

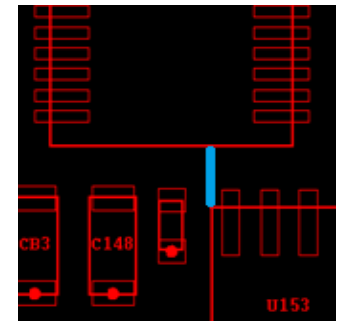
## SMT placement



Pin overhanging soldermask can cause solder bridging



Toeprint too close to conveyed edge can be damaged by de-paneling equipment

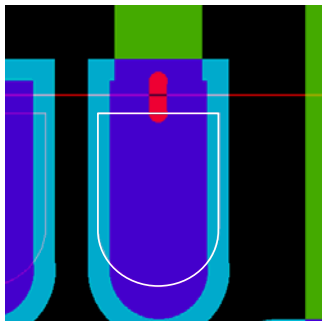


Component clearance requirements vary by type, orientation and placement machine

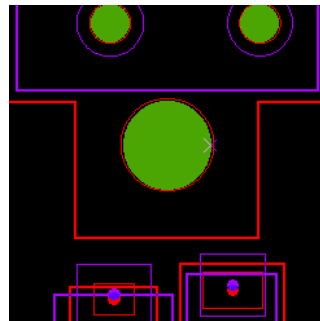


# A Contract Manufacturer's View

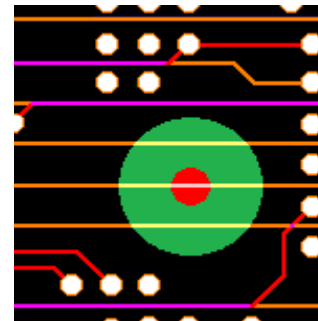
## SMT placement



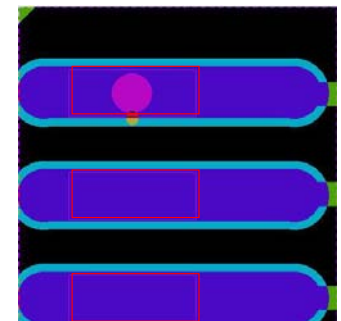
Insufficient heel distance will lead to a weak solder joint



Actual component body (red) covers mounting hole that CAD package (purple) didn't detect



Adjacent layer copper makes it difficult for SMT equipment to identify local fiducials



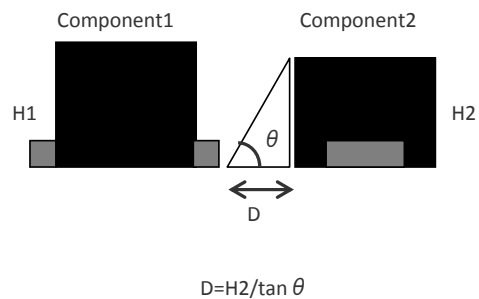
Pins too wide for pads can lead to insufficient side fillets of solder joints

# A Contract Manufacturer's View

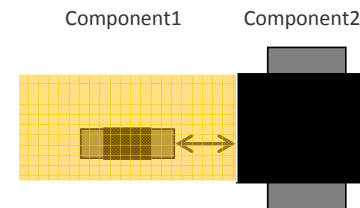
## AOI/AXI

Detect toeprints those are shadowed from AOI camera

Check with the height of component2 even if it is not higher.

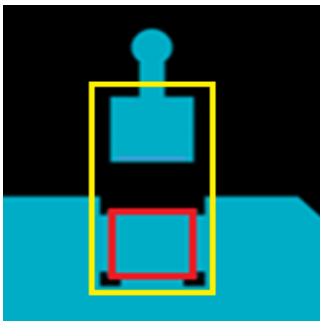


Check when toeprint of component1 is orienting component2.

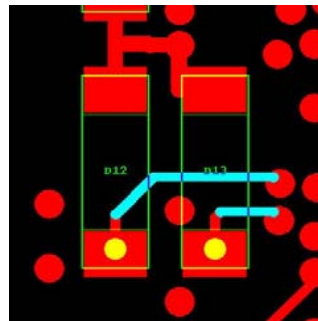


# A Contract Manufacturer's View

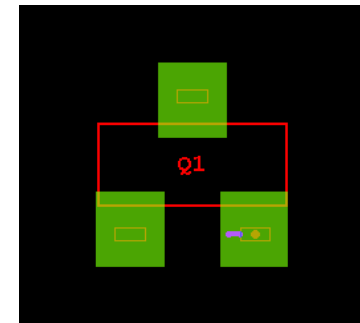
Reflow, wave & selective soldering



Different trace widths connecting pads can cause the component to tombstone during reflow



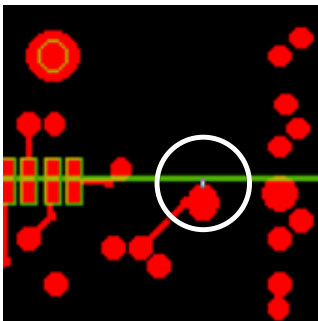
Traces under a zero-offset device may cause the component to rock, causing a poor solder joint



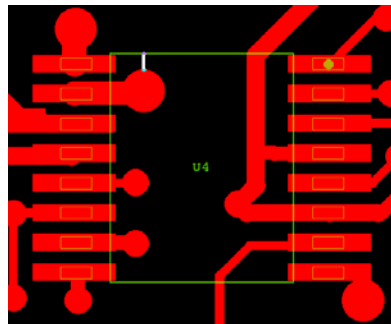
Pad is too large for small component pin. Part will twist during reflow

# A Contract Manufacturer's View

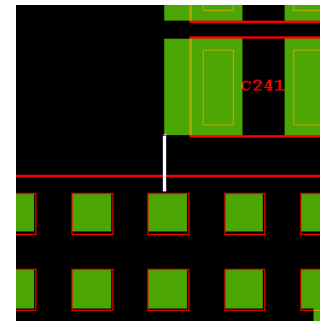
## Test & repair



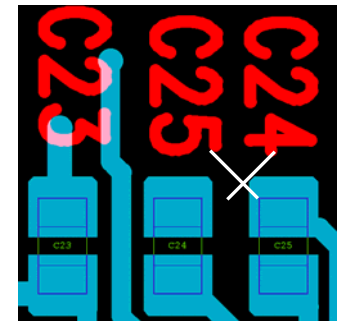
Testpoint may not be accessible with flying probe due to close proximity of a tall component



Testpoint under component is not accessible



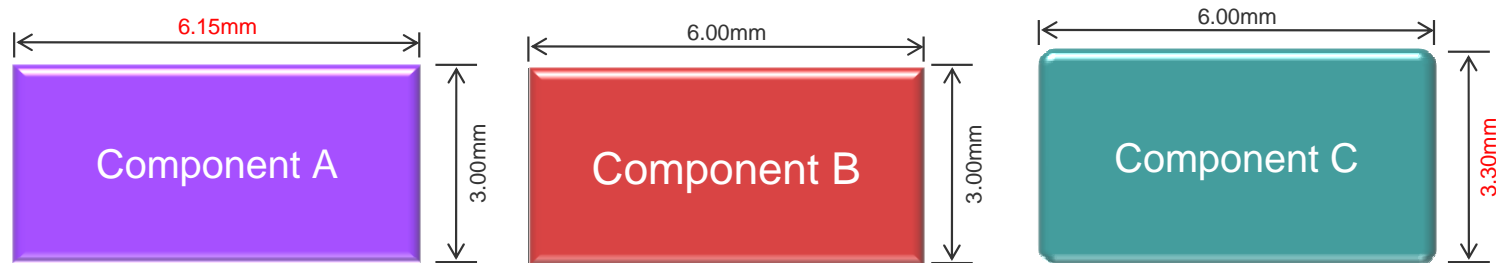
Tall components too close to adjacent components make it difficult to access the short component for repair



Misplaced reference designator causes confusion at test, repair and for manually placed components

# A Contract Manufacturer's View

Electrically equivalent materials in AVL have different dimensions and pin contact areas



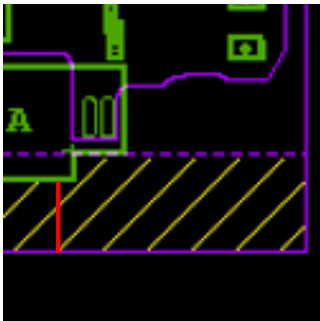
DFx Composite Body



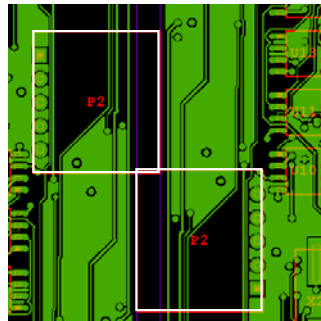
Goal: No manufacturing issues no matter which qualified parts are used

# A Contract Manufacturer's View

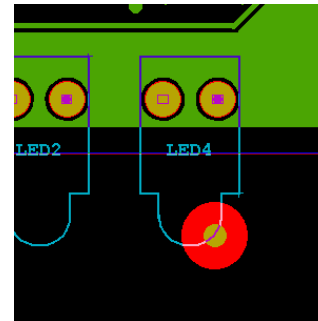
## Assembly panels



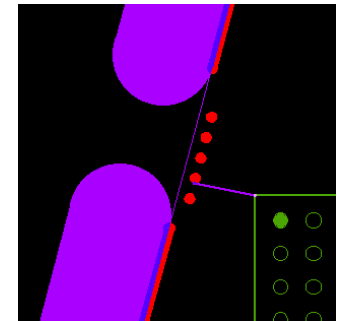
Components too close to conveyed edge can be damaged during assembly



Placement of components on adjacent PCBs cannot be done with this assembly panel design



Components protruding from edge of PCB can obscure panel fiducials



Breakaway tabs too close to SMD pad can result in damage to solder joint

# DFM Complements DRC

## DFM is Manufacturing Facing

- Checking manufacturing data
- For issues that can impact manufacturing
- Using rules aligned with your supplier's capabilities
- For chosen manufacturing processes



# DFM Complements DRC

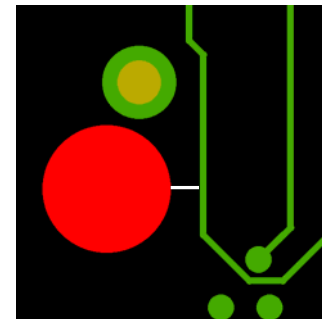
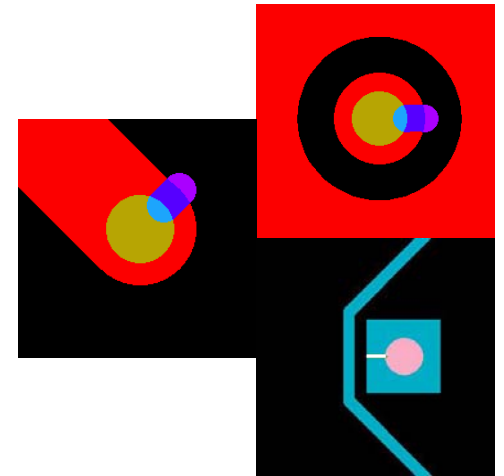
The variety of manufacturing processes requires more specific checks

## ■ Annular Ring

- DRC uses a minimum annular ring
- DFM analyzes annular ring size based on manufacturing process
  - Laser drilled via
  - Mechanically drilled TH
  - Backdrill

## ■ Drill to Copper Spacing

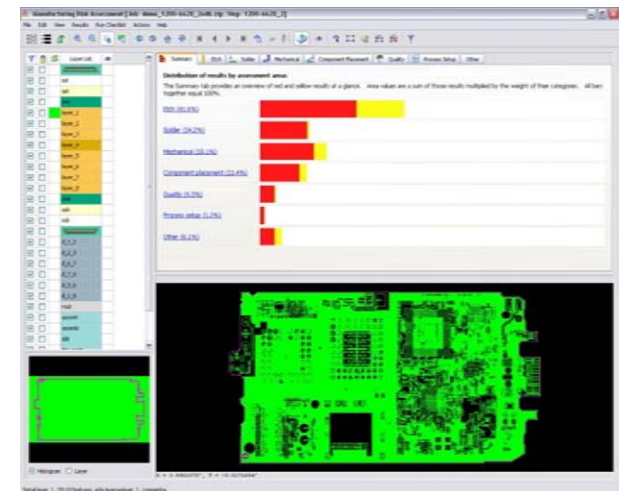
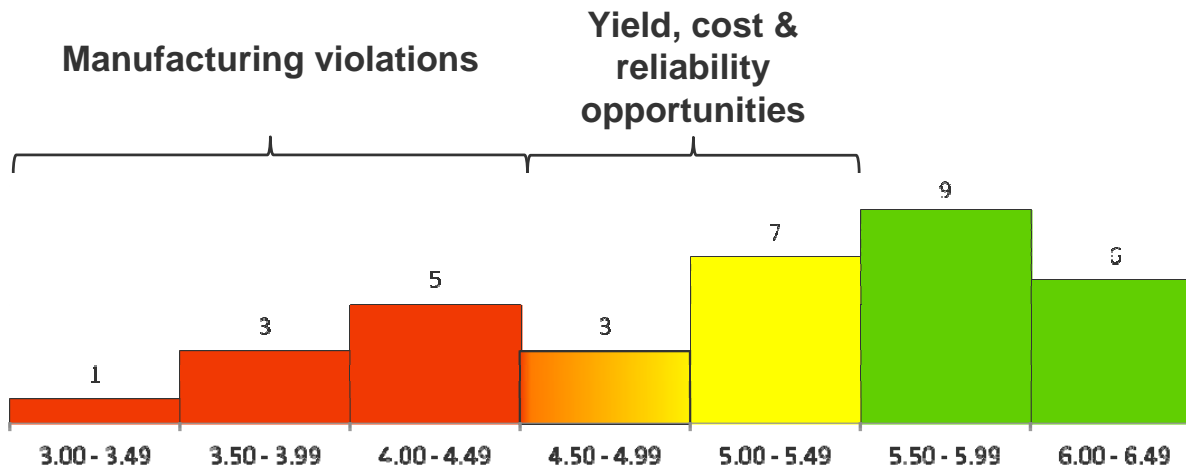
- DRC has drill to copper spacing
- DFM analyzes drill to copper spacing based on manufacturing process
- PTH can be closer to trace than NPTH



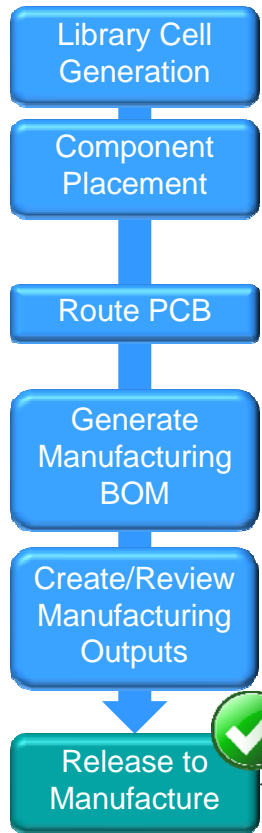


# DFM Complements DRC

- DFM enables you to optimize your design for volume manufacturing in the initial release
- An issue can pass DRC but still impact yield, cost or reliability



# Summary

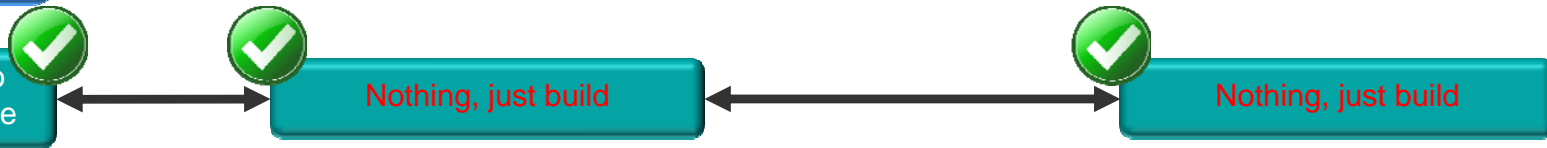


## PCB Design Review and Release Process with Design DFM

If the data used to manufacture the PCB's:

- Is reviewed against your supplier's process capabilities
- For issues that can impact manufacturing, yield, cost or reliability

What do your suppliers need to review?





[www.mentor.com/pcb](http://www.mentor.com/pcb)

## How Your Suppliers Look at Your Design